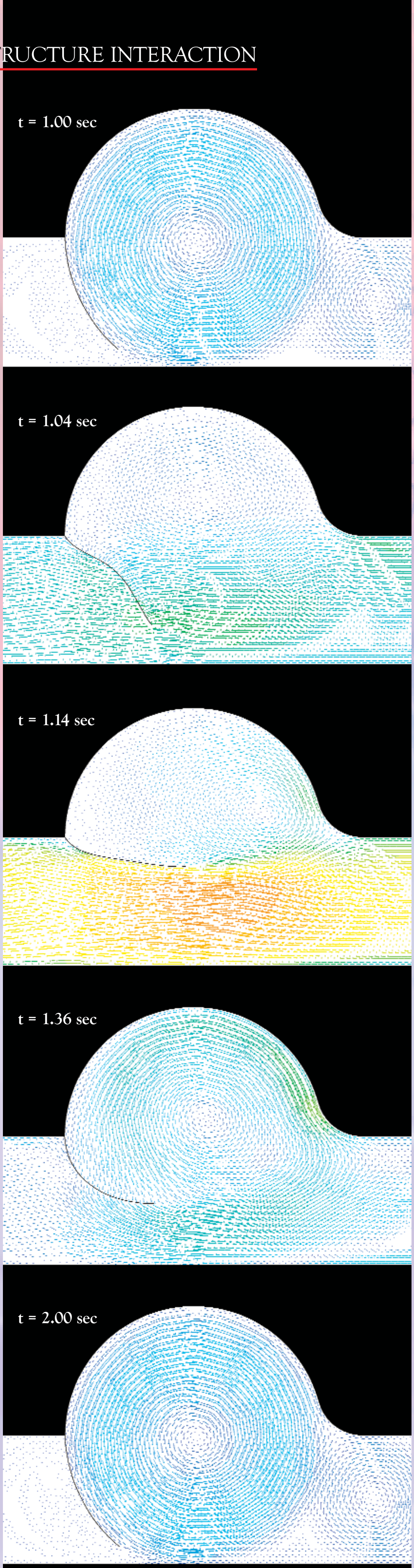

The figures show a flexible leaflet being opened by an accelerating flow; the stiffness of the leaflet is very small, so that it undergoes a huge displacement from the closed to the fully open position and back again

Artificial Heart Valve Takes Shape

By Jan Vierendeels, Joris Degroote, Kris Dumont, Lieve Lanoye, Pascal Verdonck, Ghent University, Belgium

AT GHENT UNIVERSITY, BELGIUM, research on fluid-structure interaction (FSI) modeling has been ongoing since 1996. At the beginning, in-house codes were developed to compute the interaction between the motion of the cardiac wall and the blood flow in the heart [1]. During the past ten years, CFD and CSD (computational structural dynamics) packages have made significant progress and nowadays, the best approach for coupling fluid motion and structural displacement is to use existing software packages as partitioned solvers. When using a partitioned approach, however, it can be difficult to obtain convergence, especially when the interaction between the different motions is strong. This is the case for blood flow in the heart and for blood flow through a heart valve with flexible leaflets. Both of these types of blood flow have been simulated using the FSI approach developed at Ghent University. The blood flow in the moving geometries is computed with FLUENT 6.2, and the structural problem is solved using in-house software. The coupled calculation is robust and does not cause undue convergence difficulty, even for strongly interacting system responses.

During the calculation, an implicit coupling method is used. For each new time, a new position of the boundary and the corresponding load distribution on that boundary is sought. This information is the result of simultaneous solutions of the CFD and CSD solvers, and could be obtained by alternating the solver calls. For strongly coupled cases however, such an approach can only be stabilized by reducing the underrelaxation factors or by using an Aitken-like technique, both of which tend to slow the rate of convergence.

To speed the convergence, a reduced order model is used [2]. If the Jacobian of the fluid solver were known when solving the structural problem, one would know how the load would change when a certain boundary displacement is applied, so the load change would not need to be calculated with the fluid solver. The boundary condition for the structural solver (the load) would not need to be assumed constant during this call, but could be written as a function of the unknown boundary position itself, resulting in a much better prediction for the boundary position. Since the Jacobian of the fluid solver is not available, a reduced order model of the solver can be built for which a Jacobian can be constructed. This approximation of the Jacobian of the fluid solver is then used for the coupled calculation. During each time step, whenever FLUENT is called a boundary displacement mode is applied and the corresponding load change at the boundary is retrieved through user defined functions (UDFs). This information is used to build up the reduced order model for the fluid problem. The structural solver is then called with the variable load boundary condition. FLUENT is called again and the reduced order model is updated. The loop is executed until a residual drop of four to five orders of magnitude is obtained. This new technique for FSI problems has proved to be robust and powerful, and is ideally suited for FSI problems with strong coupling. ■

References:

1 Vierendeels, J.A., Riemslagh, K., Dick, E. and Verdonck, P.R.: Computer Simulation of Intraventricular Flow and Pressure Gradients During Diastole. Journal of Biomechanical Engineering – Transactions of the ASME, 122(6):667-674, 2000.

2 Vierendeels, J.A.: Implicit Coupling of Partitioned Fluid-structure Interaction Solvers using a Reduced Order Model. In Proc. of the 35th AIAA Fluid Dynamics Conference and Exhibit, June 6-9 2005, Toronto, Ontario, Canada. AIAA-2005-5135. AIAA Meeting papers on disc, vol. 10, nr 11-12. ISBN 1-56347-763-7.